NASA Technical Memorandum 100076

Three-Dimensional Incompressible Navier-Stokes Computations of Internal Flows

D. Kwak, J. L. C. Chang, S. E. Rogers,
and M. Rosenfeld

March 1988

(NASA-TM-100076) THREE-DIMENSIONAL INCOMPRESSIBLE NAVIER-STOKES COMPUTATIONS OF INTERNAL FLOWS (NASA) 14 p CSCL 01C

N88-19425

Unclas
G3/03 0131798

National Aeronautics and
Space Administration

NASA Technical Memorandum 100076

Three-Dimensional Incompressible Navier-Stokes Computations of Internal Flows

D. Kwak, Ames Research Center, Moffett Field, California
J. L. C. Chang, Rocketdyne, Rockwell International, Canoga Park, California
S. E. Rogers, Sterling Federal Systems, Palo Alto, California
M. Rosenfeld, Ames Research Center, Moffett Field, California

March 1988

National Aeronautics and
Space Administration

Ames Research Center
Moffett Field, California 94035

THREE-DIMENSIONAL INCOMPRESSIBLE NAVIER-STOKES COMPUTATIONS OF INTERNAL FLOWS

D. Kwak,* J. L. C. Chang,† S. E. Rogers,‡ and M. Rosenfeld§

Ames Research Center

SUMMARY

Several incompressible Navier-Stokes solution methods for obtaining steady and unsteady solutions are discussed. Special attention is given to internal flows which involve distinctively different features from external flows. The characteristics of the flow solvers employing the method of pseudocompressibility and a fractional step method are briefly described. The present discussion is limited to a primitive variable formulation in generalized curvilinear coordinates. Computed results include simple test cases and internal flow in the Space Shuttle main engine hot-gas manifold.

INTRODUCTION

Incompressible internal flows are encountered in many realistic engineering problems such as the flow through impeller passages and duct flow. Geometric variation for these flows is diverse and naturally the computational approach has to take this into account. Internal flows are in general very different from the extenal flows. For instance, the boundary layer in external flow is usually very thin compared to the characterisitc length of the moving object, while in internal flow the entire flow field is likely to be viscous. The boundary layer is often separated by abrupt changes in geometry and the blockage effect caused by the separated zone becomes significant.

One of the major motivations of the present work was to develop numerical simulation capability especially suitable for internal flows in the Space Shuttle maine engine (SSME) power head. To upgrade the SSME without increasing the weight and size of the existing engine, it became essential to understand the dynamics of the hot-gas flow in the engine power head. Because of the complexity of the geometry, an experimental approach was extremely difficult as well as time consuming. Computational simulation, therefore, offers an economical alternative to complement the experimental work in analyzing the current configuration, and to suggest new, improved design possibilities. The Reynolds number of the primary flow in the power head is of order 10^6. Because of the high gas temperature, the Mach number is less than 0.12. The flow is turbulent and is practically incompressible. In the past, numerous numerical methods of simulating viscous incompressible flows have been developed (ref. 1). The present paper is a short

* Research Scientist, NASA Ames Research Center.

† Senior Staff Scientist, Rocketdyne, Rockwell International, Canoga Park, CA.

‡ Research Scientist, Sterling Federal Systems, Palo Alto, CA.

§ NRC Fellow, NASA Ames Research Center.

To be presented at the Twelfth IMACS Conference on Scientific Computation, Paris, France, July 18-22, 1988.

summary of our recent effort to develop alternative methods for simulating three-dimensional viscous incompressible flows in generalized coordinates.

This work is partially sponsored by the NASA Marshall Space Flight Center.

SOLUTION METHODS

A major problem of solving the incompressible Navier-Stokes equations comes from the lack of a pressure term in the continuity equation. In realistic three-dimensional applications, satisfying continuity in a reasonable amount of computing time becomes a primary issue in addition to accuracy and robustness. For convenience and flexibility, a primitive variable formulation in generalized curvilinear coordinates is chosen, and our discussion is limited to this formulation. In this section, three computer codes recently developed at NASA Ames Research Center and related solution methods used in the codes are summarized. Derivation of equations and algorithmic details can be found in full-length versions of the authors' publications (refs. 2-6).

Pseudocompressibility Method

Recent advances in the state of the art in computational fluid dynamics (CFD) have been made in conjunction with compressible flow computations. Therefore, it is of significant interest to be able to use some of these compressible flow algorithms. To do this, the artificial compressibility method (ref. 7) can be used. In this formulation, the continuity equation is modified by adding a time-derivative of the pressure term resulting in a hyperbolic-parabolic type of time-dependent system of equations. This method was originally intended for the steady-state computation of incompressible flows. However, by introducing a pseudotime iteration, this can be made time accurate (refs. 6 and 8).

In an incompressible flow, a disturbance in the pressure causes waves which travel with infinite speed. When pseudocompressibility is introduced, waves of finite speed result in which the magnitude of the speed depends upon the magnitude of the pseudocompressibility. In a true incompressible flow, the pressure field is affected instantaneously by a disturbance in the flow, but with pseudocompressibility, there will be a time lag between the disturbance and its effect on the pressure field (ref. 3). The interaction of the pressure wave propagation and the viscous field is especially pronounced in internal flows (ref. 3). Various applications evolved from this concept have been reported for obtaining steady-state solutions (refs. 2-4, 9 and 10).

INS3D Code By constructing a pseudocompressible form of governing equations, fast, implicit schemes developed for compressible flows, such as the approximate-factorization scheme (ref. 11) can be implemented. The present code is written for obtaining steady-state solutions. The spatial discretization uses second-order central differencing with additional numerical dissipation terms. This code has been validated and numerous three-dimensional problems have been solved using this code (refs. 2-4, 12 and 13).

INSUP Code – To obtain time-accurate solutions using the pseudocompressibility formulation, it is necessary to satisfy continuity at each time step by subiteration in pseudotime. To use a large time step in the pseudotime iteration, an upwind differencing scheme based on flux-difference splitting is used combined with an implicit line relaxation scheme. This removes the factorization error and the need for numerical dissipation terms. The code has been validated and excellent results have been obtained (ref. 6).

Fractional-Step Method

The fractional-step method can be used for time-dependent computations of the incompressible Navier-Stokes equations (refs. 5 and 14-16). Here, the discretized equations are advanced in time by decoupling the solution of the momentum equation from that of the continuity equation. The common application of this method is done by two steps. The first step is to solve for an auxiliary velocity field using the momentum equation in which the pressure-gradient term can be computed from the pressure in the previous time step (ref. 15), or can be excluded entirely (ref. 16). In the second step, the pressure which maps the auxiliary velocity onto a divergence-free velocity field is computed. Various other operator splittings can be adopted by treating the momentum equation as a combination of convection, pressure, and viscous terms.

INSFS Code – A generalized flow solver based on this approach using a staggered grid has been developed (ref. 5). The governing equaitons are discretized conservatively using finite volumes. Rather than choosing the Cartesian velocity components as dependent variables, the volume fluxes over the faces of the computational cells are used. They are equivalent to the contravariant velocity components described in a staggered grid. This procedure, combined with accurate and consistent approximations of the geometric quantities, satisfies the discretized mass conservation equation exactly. A novel four-color ZEBRA scheme is devised for solving the Poisson equation for pressure correction. Several computational results have been compared with experiments and other numerical solutions in reference 5.

COMPUTED RESULTS

Validation of Flow Solvers

The three flow solvers described above have been validated by computing various basic fluid dynamics problems. A few examples are listed in this section.

In figure 1, vortex shedding from a circular cylinder is summarized by a plot of Strouhal number as a function of the Reynolds number based on diameter of the cyinder. Strouhal number is the dimensionless frequency of the vortex shedding and a good agreement is seen between the computed results obtained by INSUP and the experiments by Kovasznay (ref. 17) and Roshko (ref. 18).

In figure 2, the time evolution of separation length for flow over a circular cylinder at Reynolds number of 40 is compared with one other computation (ref. 19) and with some experimental results (ref. 20). The computed results in figure 2 are obtained using INSFS.

SSME Flow Analysis

In the SSME staged combustion cycle, the fuel is partially burned at very high pressure and relatively high temperature in the preburners. The resulting hot gas is used to run the turbine and is then discharged from the gas turbine to the annular turnaround duct (TAD), and experiences a 180° turn before it diffuses into the fuel bowl. This assembly is called the hot-gas manifold (HGM). The SSME flow simulation was performed using INS3D code.

To simulate this flow undergoing a 180° U-turn, it is necessary to incorporate strong curvature effect in the turbulence modeling. Several levels of turbulence models have been implemented in INS3D code. However, for the present problem, a length scale determined by the point of minimum vorticity is used. This length scale is incorporated into an extended Prandtl-Karmann mixing-length theory. The combination of these automatically account for curvature effect. Full details of this model are given in references 13 and 21. In figure 3(a), the grid in the 180° turn region is shown. In figure 3(b), computed pressure coefficients for the inner and the outer walls are compared with experiment (ref. 22).

In figure 4, the computer model of the current three-duct SSME power head is shown. This model is chosen to analyze critical areas where the dynamics of the hot-gas flow is expected to have a significant effect on the overall performance of the SSME. Multiple zone computations were performed using the grid shown. From this computational flow analysis and also from experiments, the center duct of the current three-duct HGM is found to transfer a limited amount of mass flow (about 10% of the total flow). Also the transverse pressure gradient remains large together with a large bubble of separation after the 180° turn. To reduce this large separation bubble, a parametric study is performed to find the best possible configuration. To improve the quality of the flow, a large-area, two-duct design concept has been developed. In addition, the ducts are chosen to have an elliptical shape to distribute the mass flow evenly to the main injector region. A grid for a two-duct model is shown in figure 5. Computational simulation of this new two-duct configuration shows that the separation bubble existing in the present design is practically removed in the new configuration. This is confirmed by experiments.

The most significant aspect of the present study is to pinpoint the locations where flow experiences the most energy loss. An important measure of the energy losses is the mass-weighted average total pressure along the flow. Figure 6 illustrates the decreasing coefficient of the mass-weighted total pressure along the centerline of the TAD, the fuel bowl, and the transfer duct. The discontinuities shown in the figure correspond to the entrance of the duct where energy fluxes are computed over different planes. In the figure, three different HGM configurations are compared. The initial two-duct design shows 28% less total pressure drop compared to the current three-duct version. After fine-tuning the two-duct configuration computationally, the pressure drop decreased even further to 36% less than the original configuration. This final configuration is then tested using cold air flow, which shows 40% reduction in pressure loss. Further details of this work can be found in references 12, 13, and 23.

CONCLUDING REMARKS

This paper presents a summary of incompressible Navier-Stokes flow-solver development work. Computed results on several validation problems are compared with experiments and other computations. Computational results of the SSME power head are favorably compared with test data, and offer information not readily available from experiments. The results show that CFD can reduce the development time and cost by suggesting the best possible configurations for final verification by experiments. The SSME application provides an example of how the present CFD capabilities can be integrated into the aerospace design process. Further study of the SSME is in progress, and the total performance improvement will be compared in the future.

REFERENCES

1. Ferziger, J. H.: Incompressible turbulent flows. J. Comp. Phys., vol. 69, pp. 1-48, 1987.

2. Kwak,D.; Chang, J. L. C.; Shanks, S. P.; and Chakravarthy, S.: A three-dimensional incompressible Navier-Stokes flow solver using primitive variables. AIAA J., vol. 24, no. 3, pp. 390-396, 1986.

3. Chang, J. L. C.;and Kwak, D.: On the method of pseudo compressibility for numerically solving incompressible flows. AIAA Paper 84-0252, Reno, Nevada, 1984.

4. Rogers, S. E.; Chang, J. L. C.; and Kwak, D.: A diagonal algorithm for the method of pseudocompressibility. J. Comp. Phys., vol. 73, no. 2, pp. 364-379, December 1987.

5. Rosenfeld, M.; Kwak, D.; and Vinokur, M.: A Solution method for unsteady, incompressible Navier-Stokes equations in generalized coordinate systems. AIAA Paper 88-0718, AIAA 26th Aerospace Sciences Meeting, Reno, Nevada, January 11-14, 1988.

6. Rogers, S. E.; and Kwak, D.: An upwind differencing scheme for the time-accurate incompressible Navier-Stokes equations. AIAA Paper 88-2583, AIAA 6th Applied Aerodynamics Conference, Williamsberg, VA, June 6-8, 1988.

7. Chorin, A. J.: A numerical method for solving incompressible viscous flow problems. J. Comp. Phys., vol. 2, pp. 12-26, 1967.

8. Merkle, C. L.; and Athavale, M.: Time-accurate unsteady incompressible flow algorithms based on artificial compressibility. AIAA Paper 87-1137, AIAA 8th Computational Fluid Dynamics Conference, Honolulu, Hawaii, June 9-11, 1987.

9. Choi, D.; and Merkle, C. L.: Application of time-iterative schemes to incompressible flow. AIAA J., vol. 23, no. 10, pp. 1518-1524, 1985.

10. Soh, W. Y.: Developing fluid flow in a curved duct of square cross-section and its fully developed dual solutions. J. Fluid Mech., to appear in September 1988.

11. Beam, R. M.; and Warming, R. F.: An implicit finite-difference algorithm for hyperbolic systems in conservation-law form. J. of Comp. Phys., vol. 22, pp. 87-110, 1976.

12. Chang, J. L. C.; Kwak, D.; and Dao, S. C.: A three dimensional incompressible flow simulation method and its application to the Space Shuttle main engine. Part I. Laminar flow. AIAA Paper 85-0175, Reno, Nev., Jan. 1985.

13. Chang, J. L. C.;Kwak, D.; Dao, S. C.; and Rosen, R.: A three-dimensional incompressible flow simulation method and its application to the Space Shuttle main engine. Part II. Turbulent flow. AIAA Paper 85-1670, AIAA 18th Fluid Dynamics and Plasmadynamics and Laser Conference, Cincinnati, Ohio, July 16-18, 1985.

14. Chorin, A. J.: Numerical solution of Navier-Stokes equations. Mathematics of Computation, vol. 22, no. 104, pp. 745-762, 1968.

15. Dwyer, H. S.; Soliman, M.; and Hafez, M.: Time accurate solutions of the Navier-Stokes equations for reacting flows. Proceedings of the 10th International Conference on Numerical Methods in Fluid Dynamics, Beijing, China, June 1986, pp. 247-251, Springer-Verlag.

16. Kim, J.; and Moin, P.: Application of a fractional-step method to incompressible Navier-Stokes equations. J. Comp. Phys., vol. 59, no. 2, pp. 308-323, 1985.

17. Kovasznay, L. S. G.: Hot wire investigation of the wake behind cylinders at low Reynolds numbers. Proc. Roy. Soc., A, vol. 198, pp. 174-190, 1949.

18. Roshko, A.: On the development of turbulent wakes from vortes streets. NACA Rep. 1191, 1954.

19. Coutanceau, M.; and Bouard, R.: Experimental determination of the main features of the viscous flow in the wake of a circular cylinder in uniform translation. Part 2. unsteady flow. J. Fluid Mech., vol. 79, pp. 257-272, 1977.

20. Collins, W. M.; and Dennis, S. C. R.: Flow past an impulsively started circular cylinder. J. Fluid Mech. vol. 60, pp. 105-127, 1973.

21. Chang, J. L. C.; and Kwak, D.: A numerical study of turbulent internal shear layer flow in axisymmetric U-duct. AIAA Paper 88-0596, Reno, Nev., January 11-14, 1988.

22. Sharma, L.; Ostermier, B.; Nguyen, L.; Dang, P.; and O'Connor, G.: Turbulence measurements in an axisymmeteric turnaround duct air flow model. Rocketdyne Division, Rockwell International, Report RSS-8763, ATU-87-5237, October 1987.

23. Yang, R-J.; Chang, J. L. C.; and Kwak, D.: A Navier-Stokes simulation of the Space Shuttle main engine hot gas manifold. AIAA Paper 87-0368, AIAA 25th Aerospace Sciences Meeting, Reno, Nevada, January 12-15, 1987.

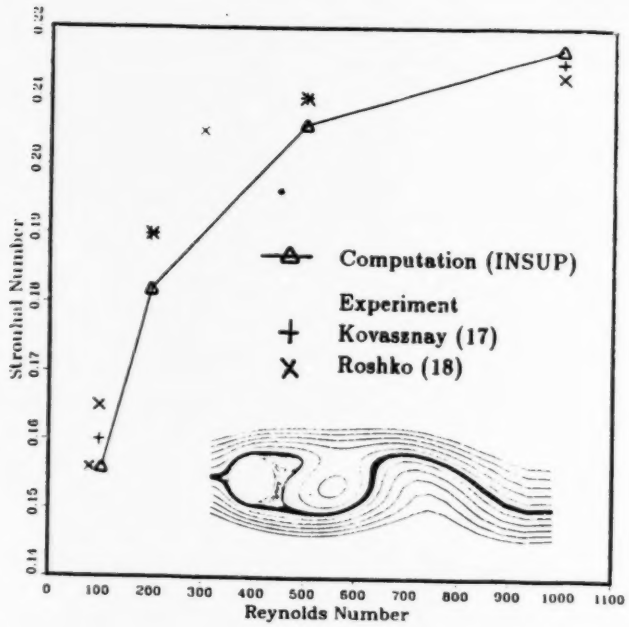

Figure 1. Vortex shedding from a circular cylinder.

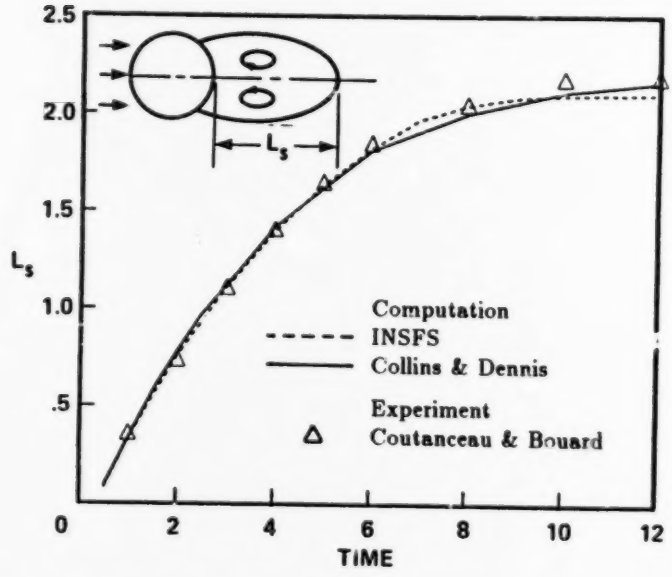

Figure 2. Time evolution of separation length for flow over a circular cylinder at R=40.

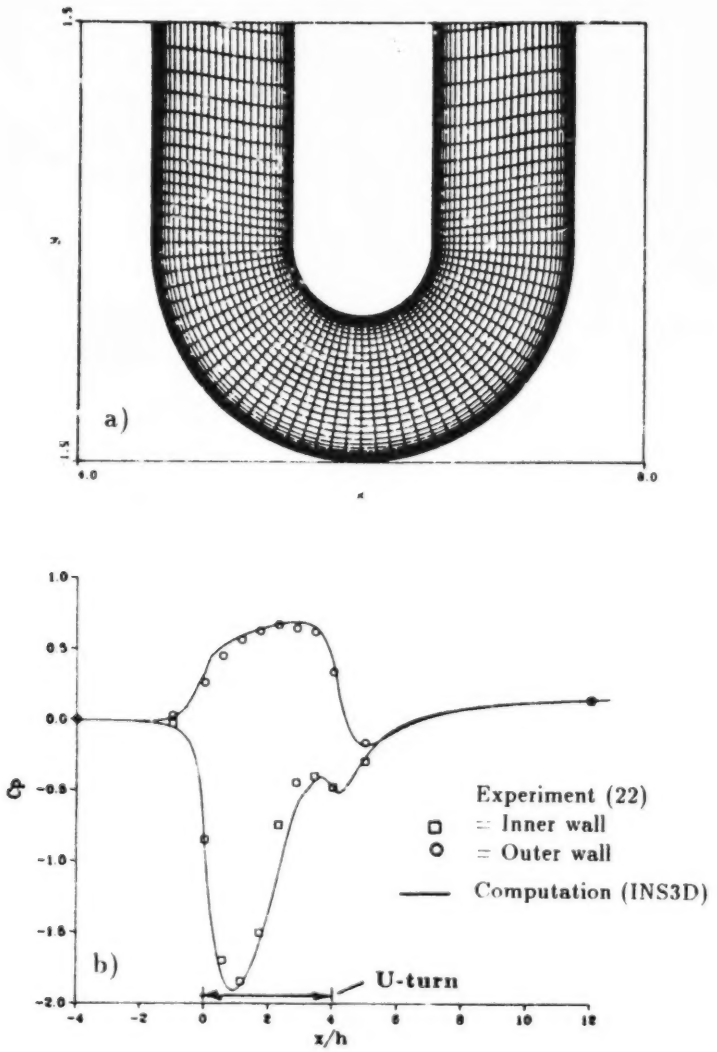

Figure 3. Flow in an axysymmetric U-duct;) grid in U-turn region, b) static pressure distribution at Re=10^5.

ORIGINAL PAGE IS
OF POOR QUALITY

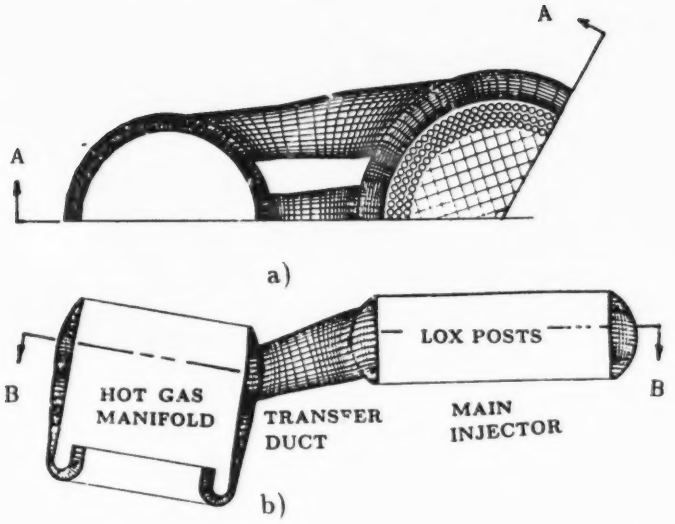

Figure 4. Grid for the current 3-duct SSME power head;
a) horizontal view (cross-section B-B),
b) vertical view (cross-section A-A).

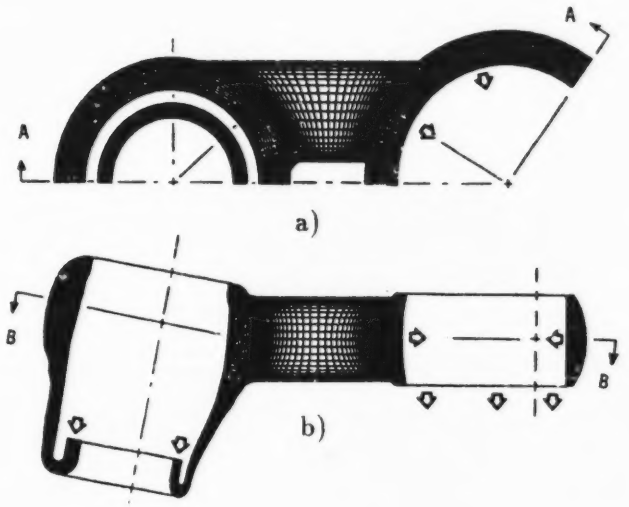

Figure 5. Grid for the new 2-duct SSME power head;
a) horizontal view (cross-section B-B),
b) vertical view (cross-section A-A).

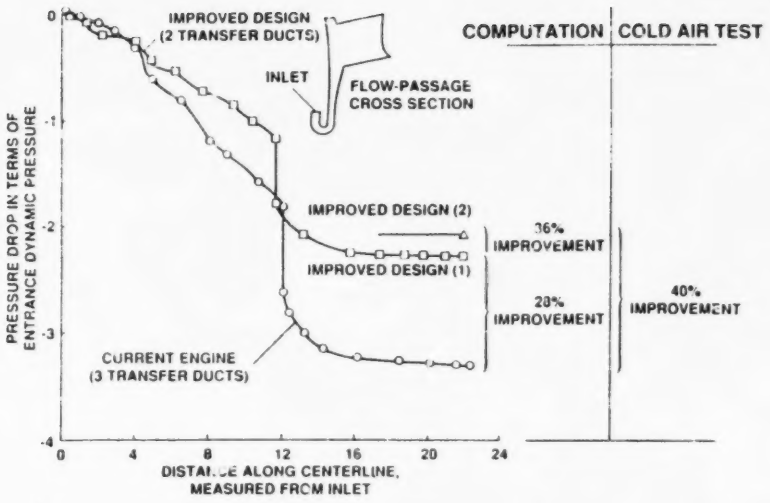

Figure 6. Comparison of pressure loss in 3-duct and 2-duct design.

NASA

Report Documentation Page

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
NASA TM-100076		

4. Title and Subtitle
Three-Dimensional Incompressible Navier-Stokes Computations of Internal Flows

5. Report Date
March 1988

6. Performing Organization Code

7. Author(s)
D. Kwak, J.L.C. Chang, S.E.Rogers, and M. Rosenfeld

8. Performing Organization Report No.
A-88111

10. Work Unit No.
505-60-01

9. Performing Organization Name and Address
NASA Ames Research Center
Moffett Field, CA 94035

11. Contract or Grant No.

13. Type of Report and Period Covered
Technical Memorandum

12. Sponsoring Agency Name and Address
National Aeronautics and Space Administration
Washington, D.C. 20546

14. Sponsoring Agency Code

15. Supplementary Notes
Point of Contact: Dochan Kwak, Ames Research Center, MS 258-1 Moffett Field, CA 94035
(415) 694-6743 or FTS 464-6743

16. Abstract

Several incompressible Navier-Stokes solution methods for obtaining steady and unsteady solutions are discussed. Special attention is given to internal flows which involve distinctively different features from external flows. The characteristics of the flow solvers employing the method of pseudocompressibility and a fractional step method are briefly described. The present discussion is limited to a primitive variable formulation in generalized curvilinear coordinates. Computed results include simple test cases and internal flow in the Space Shuttle main engine hot-gas manifold.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Incompressible Navier-Stokes equations Internal flow	Unclassified-Unlimited Subject Category - 03

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of pages	22. Price
Unclassified	Unclassified	12	A02

For sale by the National Technical Information Service, Springfield, Virginia 22161